The “Up to Next” method.

The files used for this demo are available from the Solidworks support website, it the knowledge base, under Mold design.

The background.

The development of this method came about after several molds were designed using Solidworks recommended method of core/cavity design. In it, SWX advises the user to insert a base part in a model. Next select all of the internal faces plus a user created radiated face and knit them into one face that you can use to cut block, or extrude up to.
This method works, but it has 2 major disadvantages.

1. It is fine for simple parts, but if you have a complex part with a lot of internal radii and features, it can take you a long time to pick all of the surfaces. It doesn’t work if you miss one, and I’ve spent days looking for the one fragment of a radius that is stopping the thing from working.
2. If you have to update the base part, say for an ecn change, you have to re-stitch all of the faces.

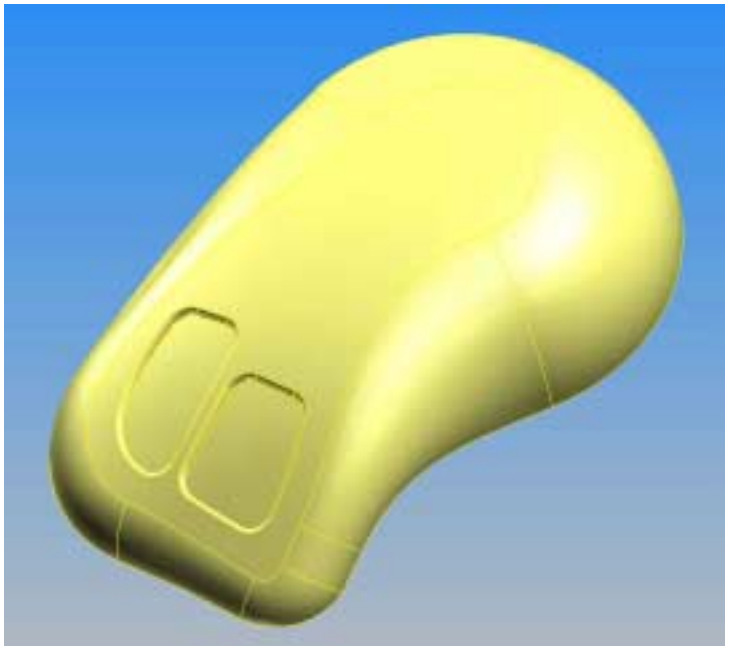
The “up to next” (utn) method does away with all of this. It takes a bit more setting up, but once it’s done, it’s a godsend.

The method.

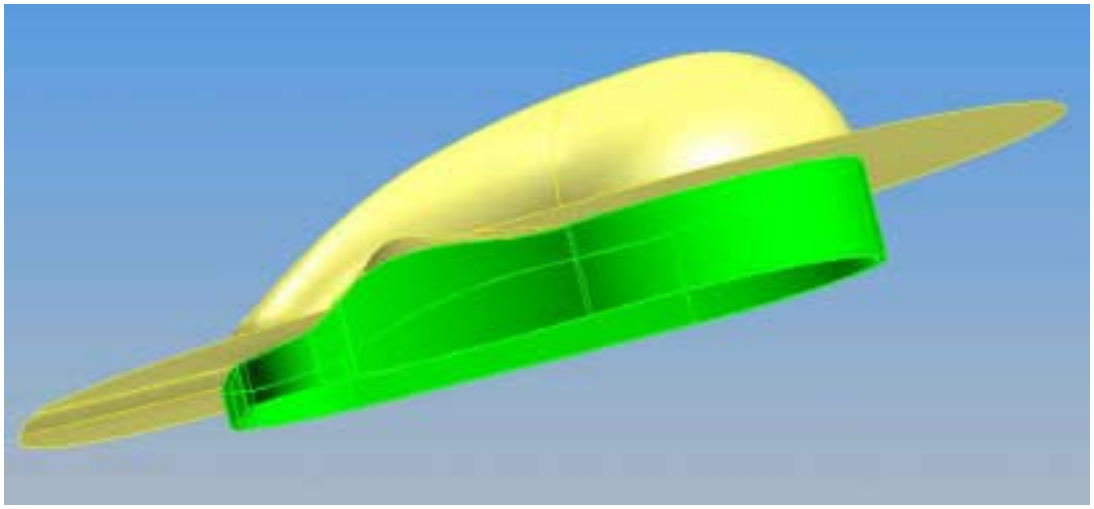
The secret behind the utn method is easy. The part should have no holes. This is fundamental to the command, and it won’t work unless it’s followed.
It may sound like a problem, but I have found that it actually helps in deciding where shut offs should occur. This in turn makes it easy to see if drafts are correct.
The easiest way to close holes is to extrude a “skin” about 0.1mm to close the hole. Do not use surfaces, as the utn command does not recognize them.

 genkimartin@mediaone.net

Example : Solidworks Mouse



The first stage is to shut off the holes. Here, because it is a solidworks native part, I was able to edit the definition of the holes, and use “Offset from surface” to create a 0.1mm “skin.” This can then be cut away after the mold has been completed.



Stage 2 is to create a radiate surface on the mouse, where you want the split line to be.
To get an easy initial split, I then extruded a boss up to the part. This gives an easily selectable straight edge to base the utn command.
Save the modified part.

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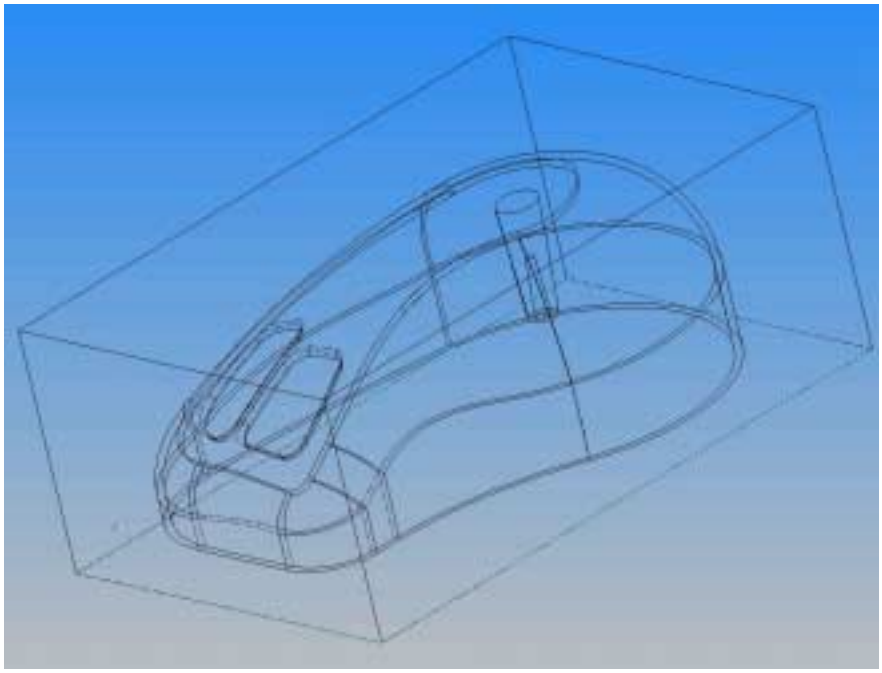
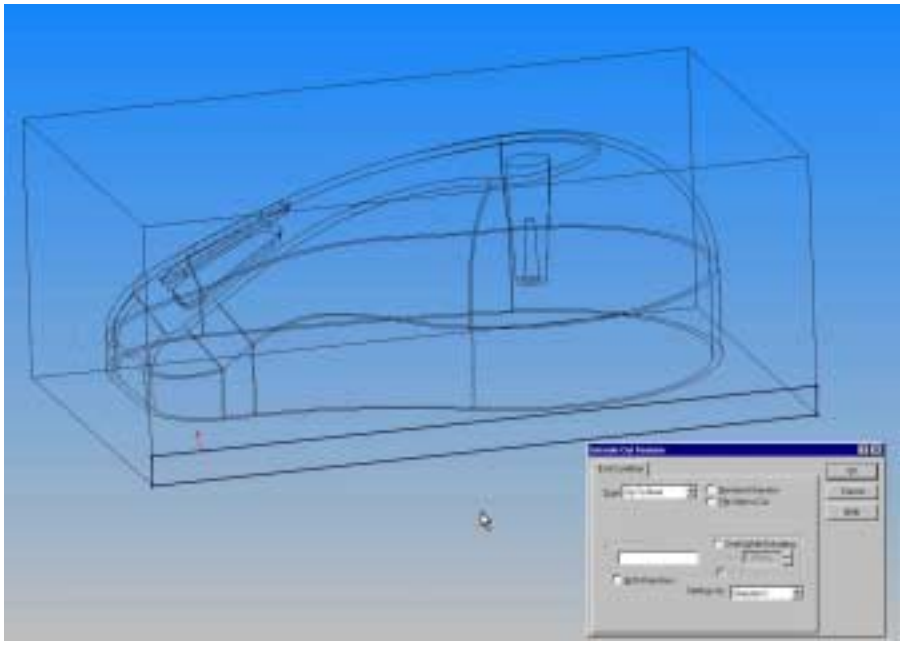


Stage 3 is to insert the modified part in a block, using the cavity command. This will provide the basis for both the core and cavity. You can save this as is and use it as a base part, or use configurations to create each.



Stage 4 is the Up to Next cut.
Select one of the faces. Place a sketch on it, and then select the straight edge created before. For the cavity, you want the cut area to be below the part. For the core, above.

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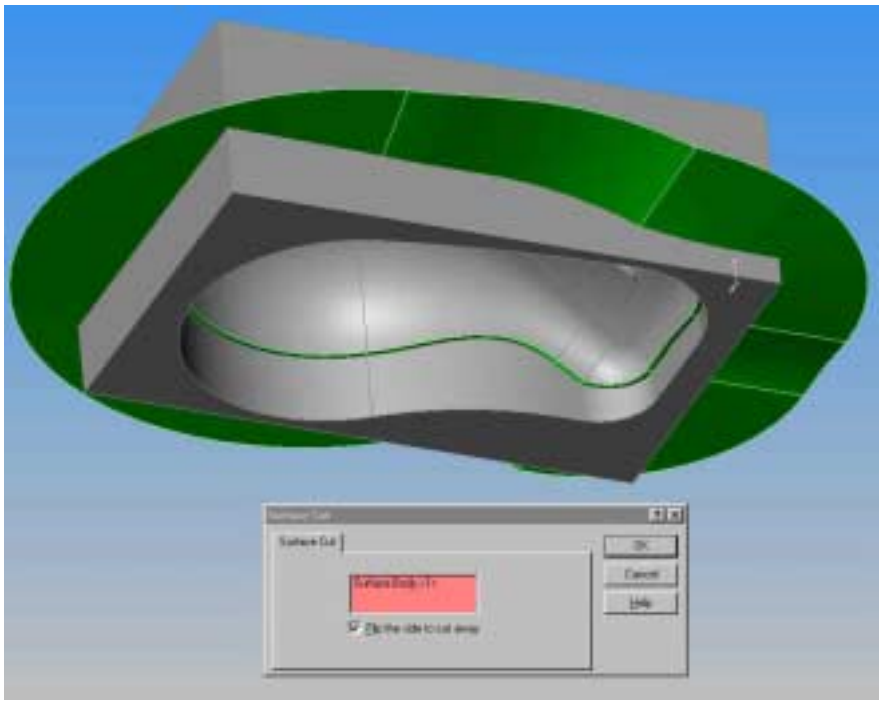
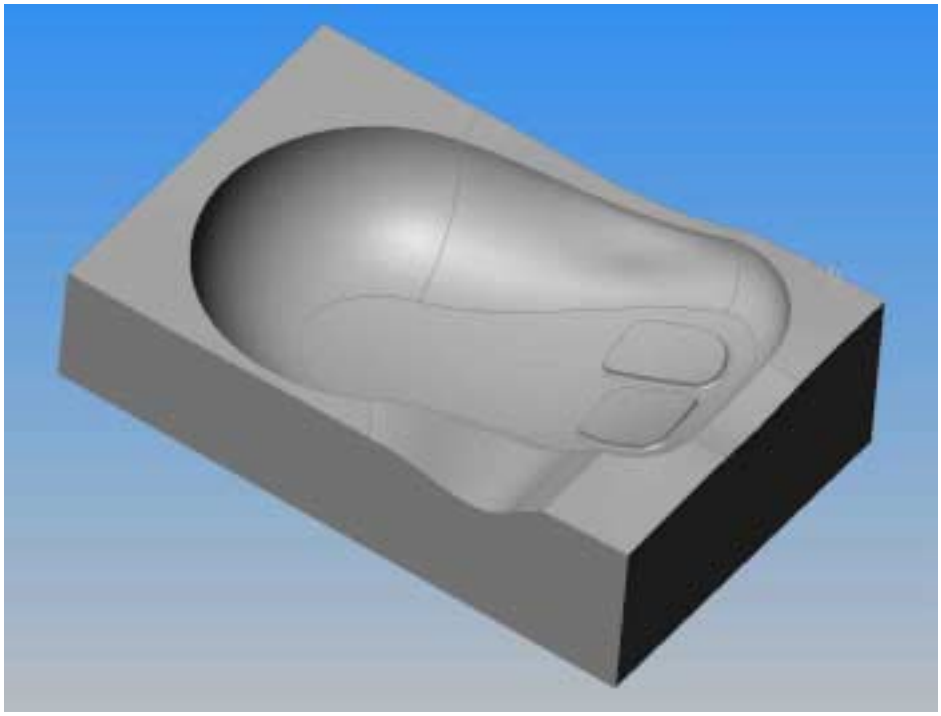


Stage 5 shows the cavity created after the utn command. I have highlighted the radiate surface which we will now use to cut away the rest of the moldbase to give us our complex parting line.



Stage 6 shows the finished cavity.

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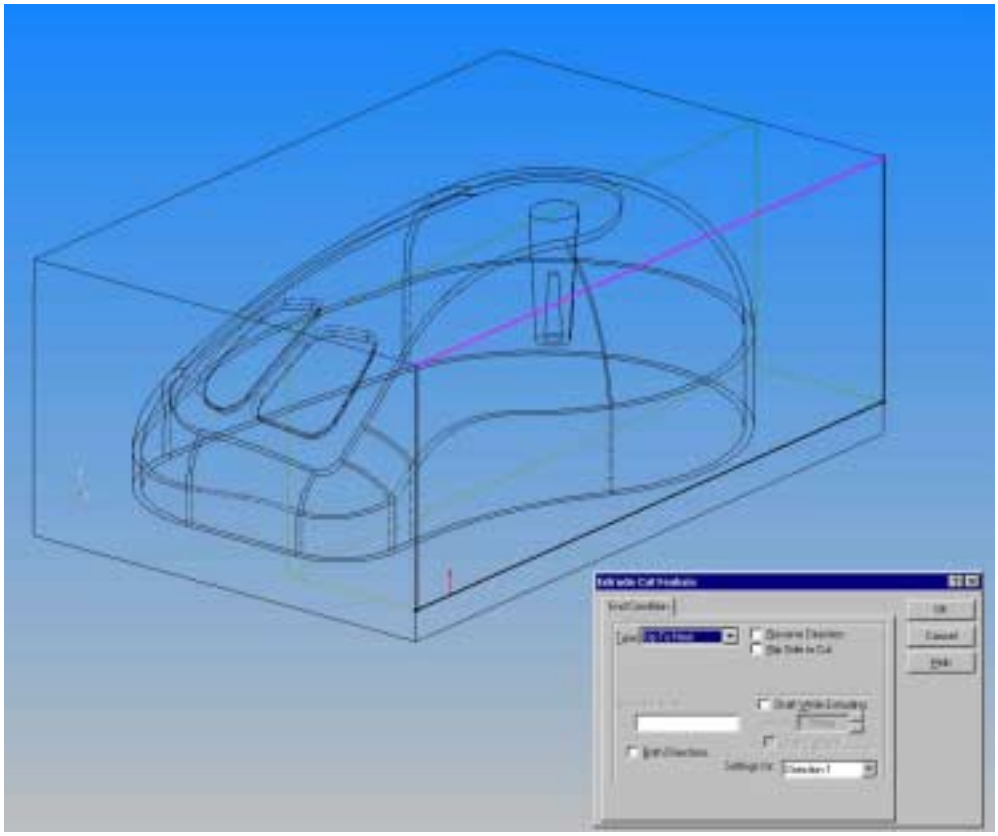
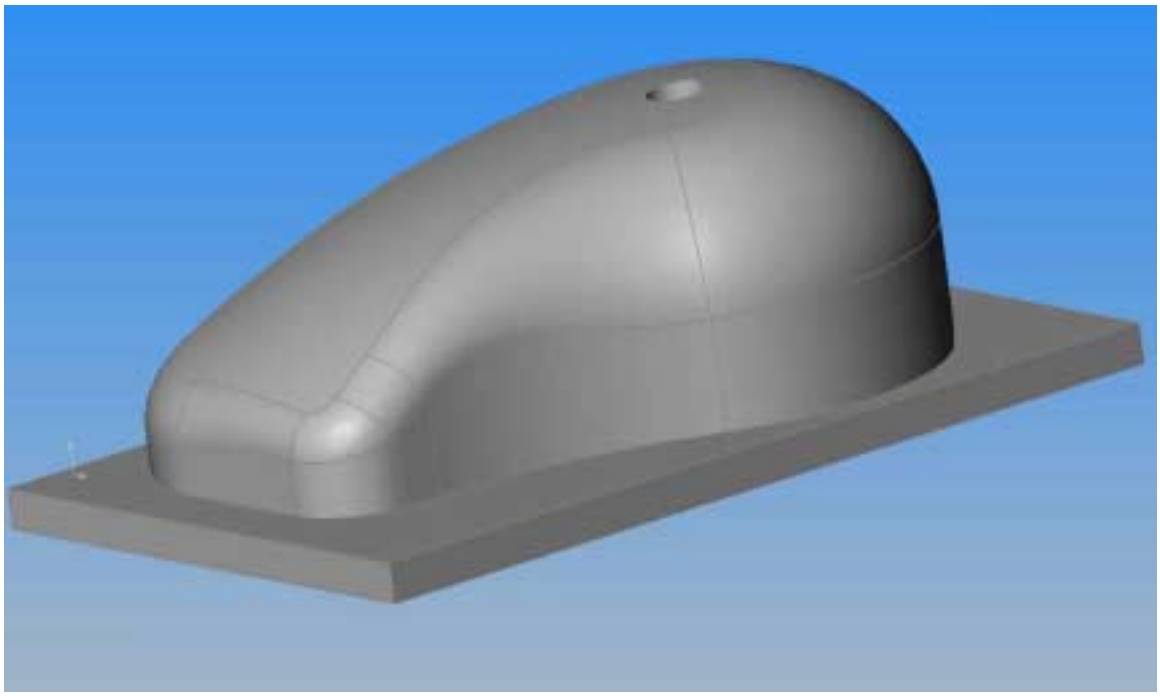
To make the core, we start with the same base part, ie a block with the cavity inserted.



We do another UTN command. This time, we select the area around the part as the area to cut away.



Here we see the core created without the complex parting line.

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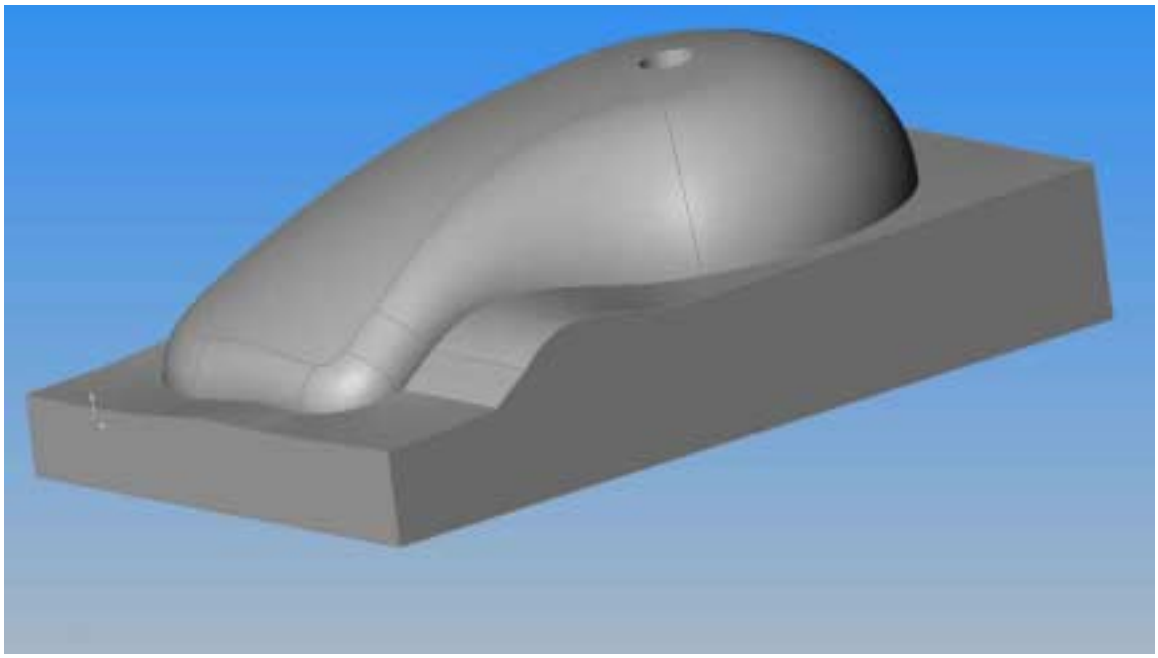
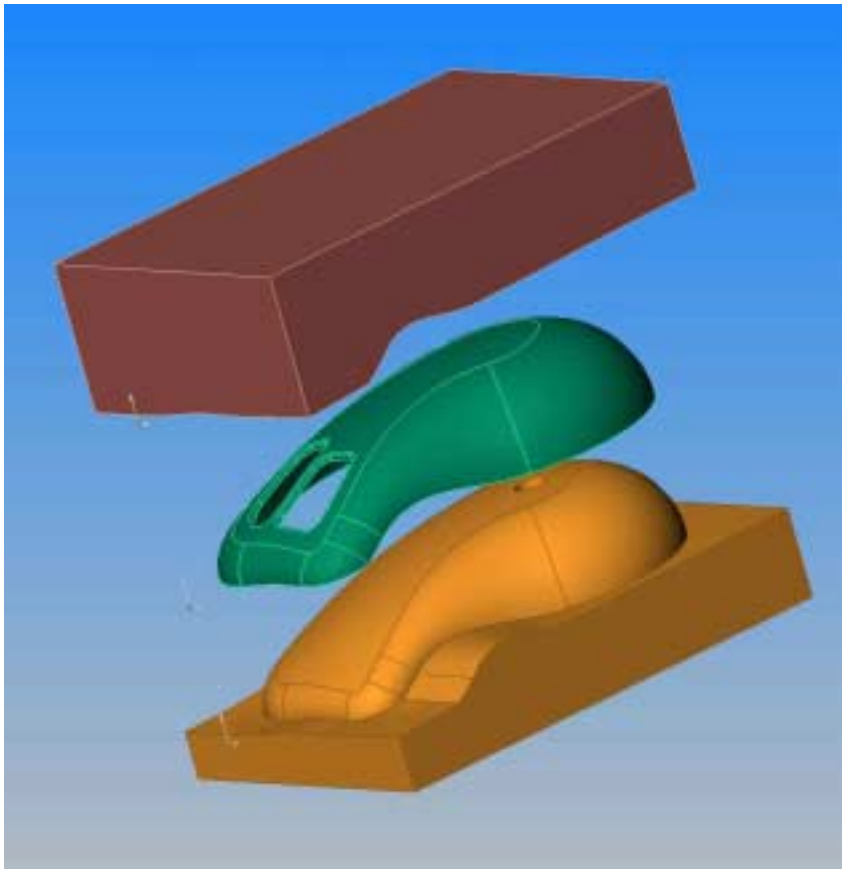


The final command is to extrude material up to the radiate surface. This then gives us our parting line.



Here we see the completed mold.

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THE SOLIDWORKS WAY
Molds/Cavities - Cutting a mold with non-planar split line
- An Advanced Example

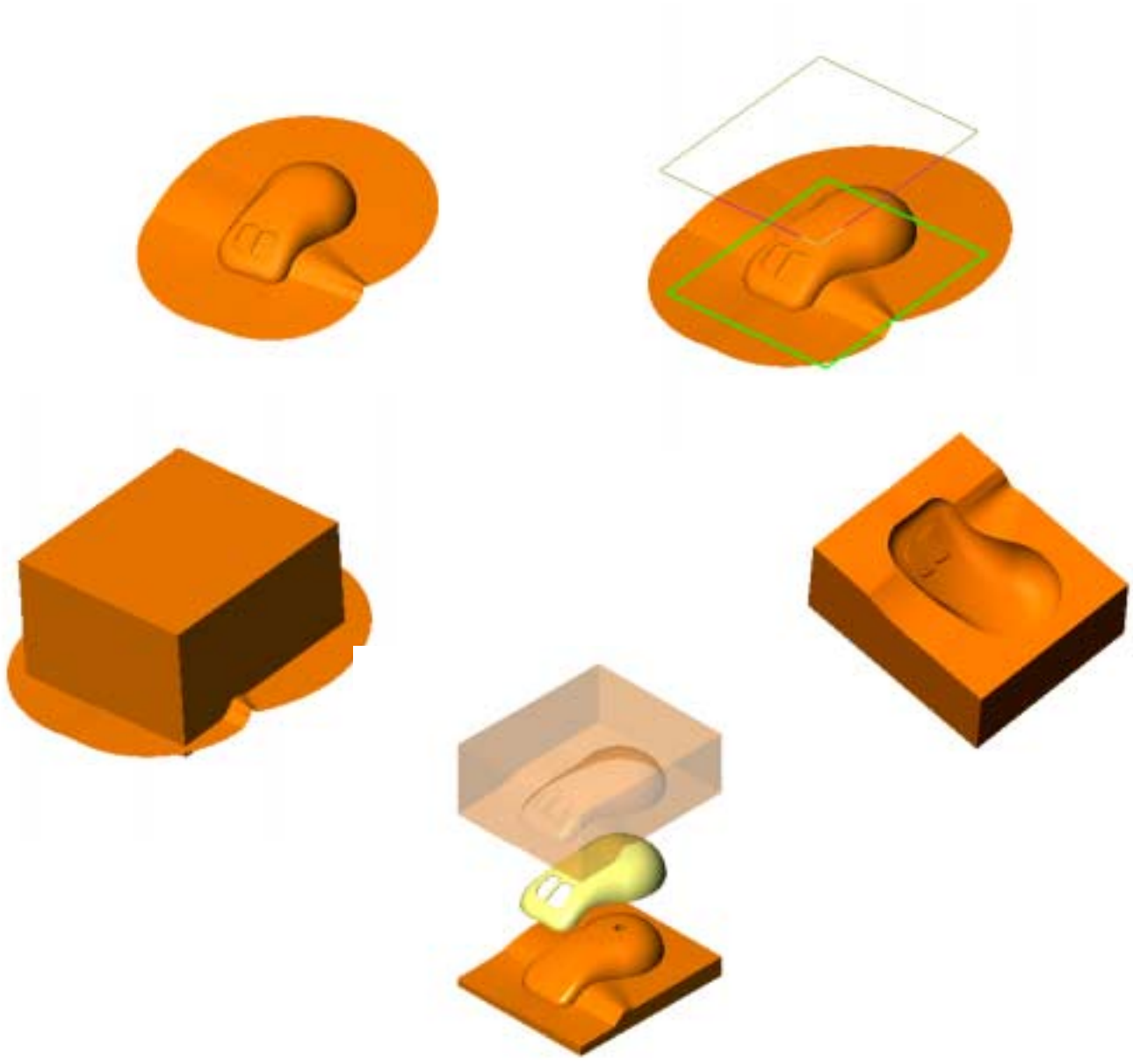
There are some cases though where we need to take a couple of extra steps to create our mold. Suppose we were to make a core of our mouse with mouse button holes. The cut with surface will not work here since the button holes would keep the mold together. We would need to make a cut to break this connectivity before we use the cut with surface using a radiating surface. This is a case where it may be advantageous to scale the part first. There are certainly other ways to do this, but you could create an entire surface to cut your mold with using the radiate surface functionality and knit surface capability available in SolidWorks. The only challenge is to create surfaces to deal with the holes. The creation of surfaces will have to be up to the designer based on his geometry and needs, but with this simple example the theory behind this example should transcend to most cases. We need to make surfaces to close off the mouse button holes. We need to decide where the mold will split. The designer will determine this and will need to select the appropriate geometry to create the surfaces that will split the mold. The first thing we are going to do is to make a derived part from our existing design. To do this we need to open a new part and then insert a base part. The base part used in the derived parts will be our designed mouse. We will do this so our mold will be parametrically defined by our original design.

We will make two of these derived parts for both parts of the mold, but not right away, we can reuse the button hole surfaces if we wait. Please also notice that the shrinkage or scale is included in our designed part. We can make configurations to suppress the scale feature when necessary. We are doing this since it easier to manage the scale from one part than from two different derived parts. We are going to create our split surfaces as if the mouse buttons were not existent on the inside surface of the mouse. The surfaces in this example were created by a lofted reference surface. A composite curve was created from many of the edges of the button hole. These edges may or may not be tangent or numerous so a composite curve is often necessary. The composite curve was created from all but one edge of the mouse button. The last edge and the composite curve were used to define the lofted reference surface. A similar approach was used to make the second lofted surface for the other mouse button hole. Depending on the surface you want to create, you may want to create more than one composite curve for lofting profiles. The holes are now considered for our mold design. We need to define the rest of the reference surface we will use to create the cut with surface. We will create a radiate surface from the inside edge of the mouse radiating outward from the mouse. The radiate distance and direction of pull will be dependent on the design. We now have three out of the four reference surfaces that are necessary to define the surface cut. We now need to knit the surfaces we have just defined with the faces of the inside of the mouse

 genkimartin@mediaone.net

cover. We will use the knit reference surface functionality to do this. At this point we have a surface, but nothing to cut it with. The next step is to create the mold body from an extrusion through the existing mouse body. This body can then be cut with our knitted surface to create our mold piece. There are many deviations from this procedure that will get you the same result. This is only to show how to get there one way. Similar steps should be used to get the other half of the mold. Save the derived part to another filename to make another copy still referencing the designed part. Now edit the surfaces to use the outside faces of the mouse cover and use the outside edge of the mouse cover to make your radiate surfaces. You can insert these three parts into an assembly on the assembly origin and fix all of them. They will aligned be correctly by default.

To download these example files please click on the link below:
http://solidweb.solidworks.com/swdocs/Support/html/KBTechTips/CuttingMolds/mouse2.zip



 genkimartin@mediaone.net

Notes.

So why change the way it's done? Simple. Any change to the part being molded means a lot of work using the solidworks method. The reason I developed this is because we wanted to add extruded part numbers to our models. The Solidworks method would have meant another 50~60 mouse clicks, plus 5 minutes re-generation time to update the surface-knit. The Up to next method updates automatically. To me there is no contest……..

I apologize in advance if there are spelling/grammar errors in this doc. It was cobbled together quickly to illustrate a point, and is very much a first draft. Comments and criticism are welcome, but don't necessarily expect an answer ☺

To get the files illustrated here, go to

http://www.myspace.com/Folders/22015753/

Use the following password to access the folder:
"solidworks" - for reading

 genkimartin@mediaone.net